CHAPTER 8

R-TYPE ELEMENTS

An R-type element is an element that imposes fixed constraints between components of motion at the grid points or scalar points to which they are connected. Thus, an R-type element is mathematically equivalent to one or more multipoint constraint equations (discussed in Chapter 7). Each constraint equation expresses one dependent degree of freedom as a linear function of the independent degrees of freedom.

In the past, and in some of the current MSC/NASTRAN documentation, all the R-type elements were referred to as rigid elements; however, the name "rigid" is misleading. The R-type elements that are rigid consist of the RROD, RBAR, RBE1, RBE2, and RTRPLT. The RBE3 and RSPLINE are interpolation elements and are not rigid. A brief description of all of the R-type elements is presented in the next section followed by a detailed description of the RBAR, RBE2, and RBE3 elements. These three elements are the most commonly used R-type elements in the MSC/NASTRAN element library.

8.1 Description of the R-Type Elements

Each of the R-type elements generates internal MPC equations in MSC/NASTRAN. These equations are generated automatically; you do not need to specify an MPC request in the Case Control Section. R-type elements are included in your model if they are included in the Bulk Data Section. (This is similar to any of the other elements in MSC/NASTRAN.) Unlike the MPC entries, an R-type element cannot be changed between subcases.

The dependent DOFs are expressed as a linear combination of the independent DOFs.

When using an R-type element, it is your responsibility to define which degrees of freedom are dependent and which are independent. The simplest way to describe this is to say that the motion of a dependent degree of freedom is expressed as a linear combination of one or more of the independent degrees of freedom. All dependent degrees of freedom are placed in what is referred to as the *m*-set. The independent degrees of freedom are temporarily placed in the *n*-set, which is the set that is not made dependent by MPCs or R-type elements. A constraint equation (an internal MPC equation) is generated for each dependent degree of freedom. A complete description of MSC/NASTRAN sets is given in Chapter 12.

Seven R-type elements are described in Table 8-1, together with the number of constraint equations generated for each of the elements. This number is the same as the number of dependent degrees of freedom that you specify for the element.

Table 8-1. R-Type Element Available in MSC/NASTRAN.

Name	Description	m = No. of Equations of Constraint Generated
RROD	Pin-ended rod which is rigid in extension.	$m = 1$
RBAR	Rigid bar with six degrees of freedom at each end.	$1 \leq m \leq 6$
RTRPLT	Rigid triangular plate with six degrees of freedom at each vertex.	$1 \leq m \leq 12$
RBE1	A rigid body connected to an arbitrary number of grid points. The independent and dependent degrees of freedom can be arbitrarily selected.	$m \geq 1$
RBE2	A rigid body connected to an arbitrary number of grid points. The independent degrees of freedom are the six components of motion at a single grid point. The dependent degrees of freedom at the other grid points all have the same user-selected component numbers.	$m \geq 1$
RBE3	Defines a constraint relation in which the motion at a "reference" grid point is the least square weighted average of the motions at other grid points. The element is useful for "beaming" loads and masses from a "reference" grid point to a set of grid points.	$1 \leq m \leq 6$
RSPLINE	Defines a constraint relation whose coefficients are derived from the deflections and slopes of a flexible tubular beam connected to the referenced grid points. This element is useful in changing mesh size in finite element models.	$m \geq 1$

The RROD element is rigid along the axis of the element only.

As an introduction to the R-type elements, consider the RROD rigid element. For the RROD element, you specify a single component of translation at one of its two end points as a dependent degree of freedom. The equivalent component at the other end is the independent degree of freedom. Consider the example shown in Figure 8-1.

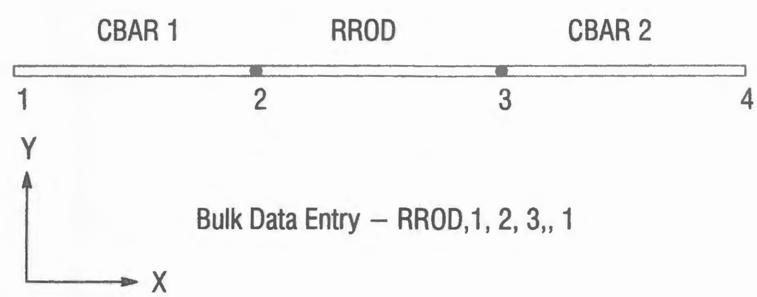


Figure 8-1. An RROD Connection.

For this example, a rigid connection is made between the X component of grid point 3 to the X component of grid point 2. When you specify the RROD entry as shown, you are placing component 1 of grid point 3 (in the global system) into the *m*-set. The remaining 5 components at grid point 3 and all the components at grid point 2 are placed in the *n*-set and hence, are independent. CBAR 1 is not connected to components Y, Z, R_x, R_y, and R_z of grid point 3 in any manner; therefore, the ends of the bars are free to move in any of these directions. However, the ends of the two CBAR elements are rigidly attached in the X-direction. Having the connection only in the X-direction is consistent with Table 8-1, which shows that one degree of freedom is placed in the *m*-set for each RROD element.

For the other six R-type elements, several degrees of freedom may be specified as members of the *m*-set or the *n*-set. (Again, the latter designation may be temporary; they may be removed by additional constraints in your model.) For the RTRPLT, RBE1, RROD, and RBE3 elements, any unlisted degrees of freedom at the grid points to which the element is joined are not connected to the element. This lack of connection can be regarded as either a sliding release or a rotating joint release, or both.

The RBAR, RTRPLT, RBE1, and RBE2 element must have exactly six independent DOFs.

Four of the R-type elements (RBAR, RTRPLT, RBE1, and RBE2) must have exactly six components of motion in the *n*-set (i.e., independent degrees of freedom). These six degrees of freedom must be able to represent all of the rigid body motions of the element. Another important requirement is that the three rules described in Chapter 7 for MPC entries must also be observed by the R-type elements. Typical applications that use R-type entries are shown in Table 8-2.

Table 8-2. Typical Application for Rigid Element.

Application	R-Type Entries
Triangular Bell Crank	RTRPLT
Rigid Engine Blocks	RBE1
Tripod with Hinged Rigid Legs	RROD
Rigid Bulkhead	RBE2
Evaluation of Resultant Loads	RBE2
Connection of a Bar Element to a Shell	RBE2 or RBE3
Hinge Between Two Plates	RBAR
Recording Motion in a Nonglobal Direction	RBAR
Relative Motion	MPC
Incompressible Fluid in an Elastic Container	MPC
"Beaming" Loads and Masses	RBE3
Change in Mesh Size	RSPLINE

One last word of caution is that the connection of two or more rigid elements to the same grid point should be done carefully to avoid specifying a degree of freedom as a member of the *m*-set more than once; otherwise, a fatal message is issued.

The next section is devoted to the RBAR element, one of the most commonly used R-type elements in the MSC/NASTRAN element library.

8.4 The RBE3 Element

The RBE3 element does not add any additional stiffness to your structure.

The RBE3 element is a powerful tool for distributing applied loads and mass in a model. the RBAR and RBE2 discussed in the previous sections, the RBE3 does not add add stiffness to your structure. Forces and moments applied to reference points are distribute set of independent degrees of freedom based on the RBE3 geometry and local weight factc

The format of the RBE3 entry is as follows:

RBE3

1	2	3	4	5	6	7	8	9
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2
	G1,3	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3
	G3,1	G3,2	-etc.-	WT4	C4	G4,1	G4,2	-etc.-
	"UM"	GM1	CM1	GM2	CM2	GM3	CM3	
		GM4	CM4	GM5	CM5	-etc.-		

Field	Contents
EID	Element identification number. Unique with respect to other elements.
REFGRID	Reference grid point identification number.
REFC	Component numbers at the reference grid point.
WTi	Weighting factor for components of motion on the following entry ; points Gi,j.
Ci	Component numbers with weighting factor WTi at grid points Gi,j.
Gi,j	Grid points whose components Ci have weighting factor WTi averaging equations.
"UM"	Indicates the start of the degrees of freedom belonging to the *m*-sel default action is to assign only the components in REFC to the *m*-sel
GMi	Identification numbers of grid points with degrees of freedom *m*-set.
CMi	Component numbers of GMi to be assigned to the *m*-set.

The manner in which the forces are distributed is analogous to the the classical bolt p analysis. Consider the bolt pattern shown in Figure 8-8 with a force and moment M act reference point A. The force and moment can be transferred directly to the weighted cer gravity location along with the moment produced by the force offset.

The RBE3 elements distribute the force to the independent point in a manner similar to a classical bolt pattern.

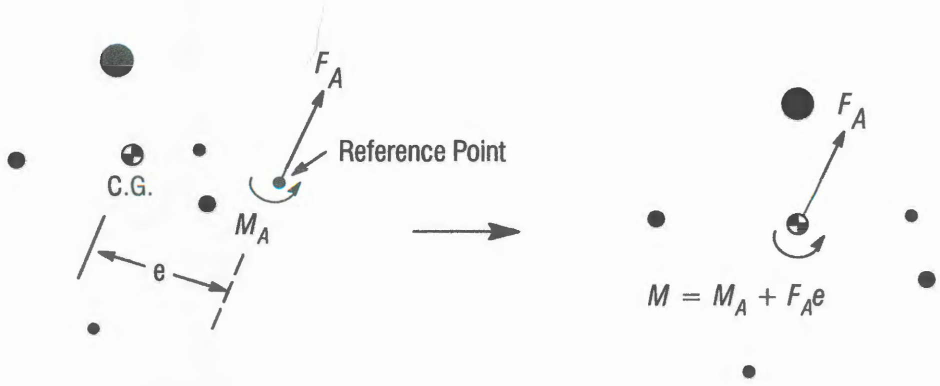
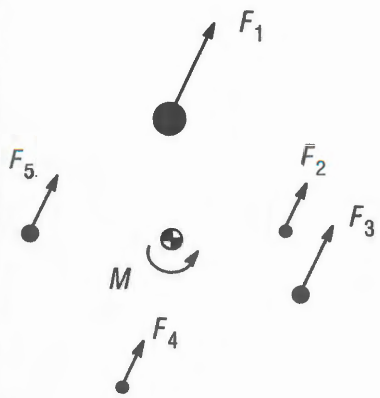
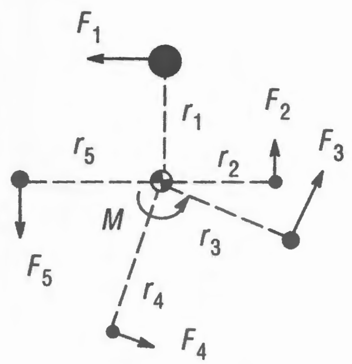


Figure 8-8. RBE3 Equivalent Force and Moment at the Reference Point.

The force is distributed to the bolts proportional to the weighting factors. The moment is distributed as forces, which are proportional to their distance from the center of gravity times their weighting factors, as shown in Figure 8-9. The total force acting on the bolts is equal to the sum of the two forces. These results apply to both in-plane and out-of-plane loadings.



$$\text{where } F_i = F_A\left[\frac{\omega_i}{\sum \omega_i}\right]$$



$$\text{where } F_i = \left(\frac{M\omega_i\, r_i}{\omega_1 r_1^2 + \omega_2 r_2^2 + \omega_3 r_3^2 + \omega_4 r_4^2 + \omega_5 r_5^2}\right)$$

where F_i = force at DOF i

ω_i = weighting factor for DOF i

r_i = radius from the weighted center of gravity to point i

Figure 8-9. RBE3 Force Distribution.

EXAMPLE

As an example, consider the cantilever plate modeled with a single CQUAD4 element shown in Figure 8-10. The plate is subjected to nonuniform pressure represented by a resultant force acting at a distance of 10 mm from the center of gravity location. The simplest way to apply the pressure is to use an RBE3 element to distribute the resultant load to each of the four corner points.

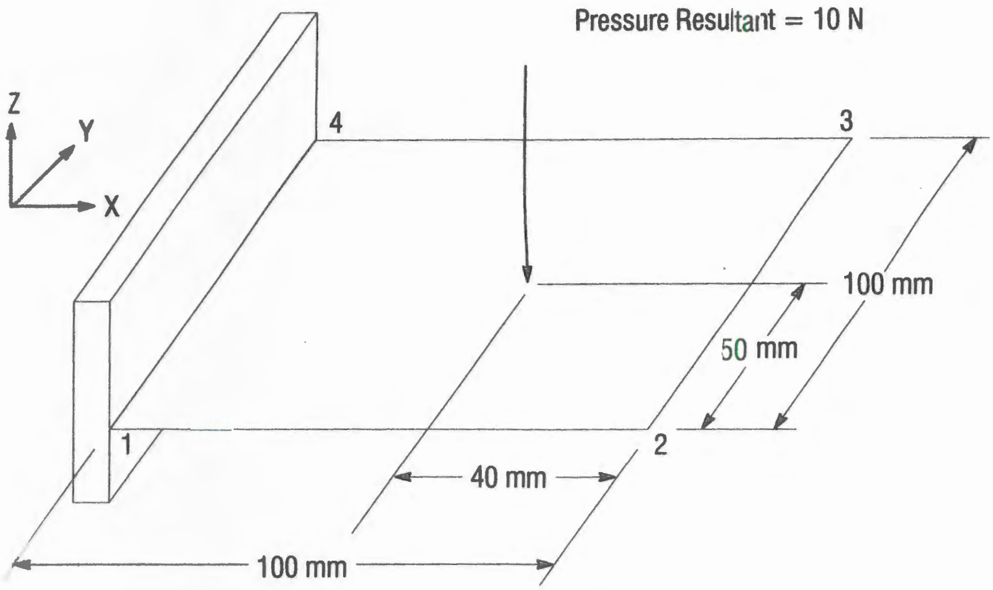


Figure 8-10. Using an RBE3 to Represent a Nonuniform Pressure Load.

The REFC components of the REFGRID point are dependent degrees of freedom.

The input file representing this example is shown in Listing 8-4. Grid point 99 is called the REFGRID and is the location where the force is applied. This point is connected only to those degrees of freedom listed on the REFC field (the T3 component in this example). The default action of this element is to place the REFC degrees of freedom in the *m*-set. The element has provisions to place other DOFs in the *m*-set instead. However, this is an advanced feature and is beyond the scope of this user's guide. The groups of connected grid points begin in field 5. For this example, the connected grid points are the corner points.

Listing 8-4. Distributing Force with an RBE3.

```
$
$ FILENAME - RBE3.DAT
$
ID      LINEAR,RBE3
SOL     101
TIME    5
CEND
TITLE = SINGLE ELEMENT WITH RBE3
SPC  = 1
LOAD = 1
OLOAD      = ALL
GPFORCE    = ALL
SPCFORCES = ALL
BEGIN BULK
$
RBE3    10              99      3       1.0     123     1       2
        3       4
FORCE   1       99              100.    0.      0.      1.
$
PARAM   POST    0
$
GRID    1               0.      0.      0.
GRID    2               100.    0.      0.
GRID    3               100.    100.    0.
GRID    4               0.      100.    0.
GRID    99              60.     50.     0.
$
PSHELL  1       4       10.     4
$
MAT1    4       4.E6            0.
$
CQUAD4  1       1       1       2       3       4
$
SPC1    1       123456  1       4
ENDDATA
```

Setting the weighting factors to be equal results in force distribution based exclusively on the spatial location of the connected grid points.

The start of a group is indicated by a real number WTi, which is used as a weighting factor for the grid points in the group. In this example, a simple distribution based only on the geometry of the RBE3 is desired so that a uniform weight is applied to all points. The weighting factors are not required to add up to any specific value. For this example, if the WT1 field is 4.0 instead of 1.0, the results will be the same.

The independent degrees of freedom for the group are listed in the Ci field. Note that all three translational DOFs are listed even though the REFC field does not include the T1- and T2-direction. All three translational DOFs in the Ci field are included because the DOFs listed for all points must be adequate to define the rigid body motion of the RBE3 element even when the element is not intended to carry loads in certain directions. If any translational degrees of freedom are not included in C1 in this example, a fatal message is issued.

The element described by this RBE3 entry does not transmit forces in the T1- or T2-direction. The two reasons for this are that the reference grid point is not connected in this direction and all of the connected points are in the same plane. Note that the rotations are not used for the independent DOFs. In general, it is recommended that only the translational components be used for the independent degrees of freedom.

A selected portion of the output file produced by this example is shown in Figure 8-11.

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	0.0	0.0	0.0
2	G	0.0	0.0	6.000000E-04	5.770412E-21	-9.000000E-06	0.0
3	G	0.0	0.0	6.000000E-04	-1.736418E-20	-9.000000E-06	0.0
4	G	0.0	0.0	0.0	0.0	0.0	0.0
99	G	0.0	0.0	3.600000E-04	0.0	0.0	0.0

L O A D   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
99	G	0.0	0.0	1.000000E+02	0.0	0.0	0.0

F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	-5.000000E+01	7.275958E-12	3.000000E+03	0.0
4	G	0.0	0.0	-5.000000E+01	7.275958E-12	3.000000E+03	0.0

G R I D   P O I N T   F O R C E   B A L A N C E

POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	0.0	0.0	-5.000000E+01	7.275958E-12	3.000000E+03	0.0
1	1	QUAD4	0.0	0.0	3.000000E+01	-1.051603E-11	-3.000000E+03	0.0
1		*TOTALS*	0.0	0.0	-2.000000E+01	-3.240075E-12	0.0	0.0
2	1	QUAD4	0.0	0.0	-3.000000E+01	1.705303E-12	7.275958E-12	0.0
2		*TOTALS*	0.0	0.0	-3.000000E+01	1.705303E-12	7.275958E-12	0.0
3	1	QUAD4	0.0	0.0	-3.000000E+01	7.673862E-12	-7.275958E-12	0.0
3		*TOTALS*	0.0	0.0	-3.000000E+01	7.673862E-12	-7.275958E-12	0.0
4		F-OF-SPC	0.0	0.0	-5.000000E+01	7.275958E-12	3.000000E+03	0.0
4	1	QUAD4	0.0	0.0	3.000000E+01	-1.068656E-11	-3.000000E+03	0.0
4		*TOTALS*	0.0	0.0	-2.000000E+01	-3.410605E-12	3.637979E-12	0.0
99		APP-LOAD	0.0	0.0	1.000000E+02	0.0	0.0	0.0
99		*TOTALS*	0.0	0.0	1.000000E+02	0.0	0.0	0.0

Figure 8-11. Select Output for the RBE3 Example.

The displacement of grid points 1 and 4 is zero due to the SPC applied to these points. The sum of the SPC forces at these two grid points is equal to the load applied to the reference grid point. The load transmitted to the corner points can be seen by inspecting the GPFORCE output. The force applied to the points due to the R-type elements and MPC entries is not listed specifically in the GPFORCE output. These forces show up as unbalanced totals (which should typically be equal to numeric zero). The forces applied to the corner grid points 1 through 4 are -20, -30, -30, and -20 N, respectively.

Typically only the components 123 are placed in the REFC field.

The most common usage of the RBE3 element is to transfer motion in such a way that all six DOFs of the reference point are connected. In this case, all six components are placed in the REFC field, and only components 123 are placed in the Ci field.

The load distributing capability of the RBE3 element makes it an ideal element to use to apply loads from a coarse model (or hand calculation) onto a detailed model of a component. For example, the shear distribution on a cross section is a function of the properties of that section. This shear loading may be applied to a cross section by performing a calculation of the shear distribution based on unit loading and using an RBE3 element with appropriate weighting factors for each grid point. In this manner, only one shear distribution need be calculated by hand. Since there are usually multiple loading conditions to be considered in an analysis, they may be applied by defining different loads to the dependent point on the RBE3 element.

EXAMPLE

For example, consider the tube attached to a back plate as shown in Figure 8-12. Suppose that you are not particularly interested in the stress in the tube or the attachment, but you are concerned about the stresses in the back plate. For this reason, you choose not to include the tube in the model; however, you want the load transferred from the tube into the back plate attachment to be approximately correct. The question is: How should the loads be applied to the attachment to simulate the behavior of the tube?

Engineering principles dictate that the Z forces (forces acting normal to the back plate) acting on the attachment vary linearly as a function of the distance from the neutral axis. The simplest method of distributing the Z forces to the attachment grid is with an RBE2 or an RBE3 element. If an RBE2 is used, the attachment ring is rigid in the Z-direction. If an RBE3 element is used, no additional stiffness is added to the attachment ring. It is an engineering decision regarding which element to use since both are approximations. For this example, use the RBE3 element. Since the weight factors for the grid points in the Z-direction are equal, the forces are distributed to the grid points based on the geometry of the grid pattern only.

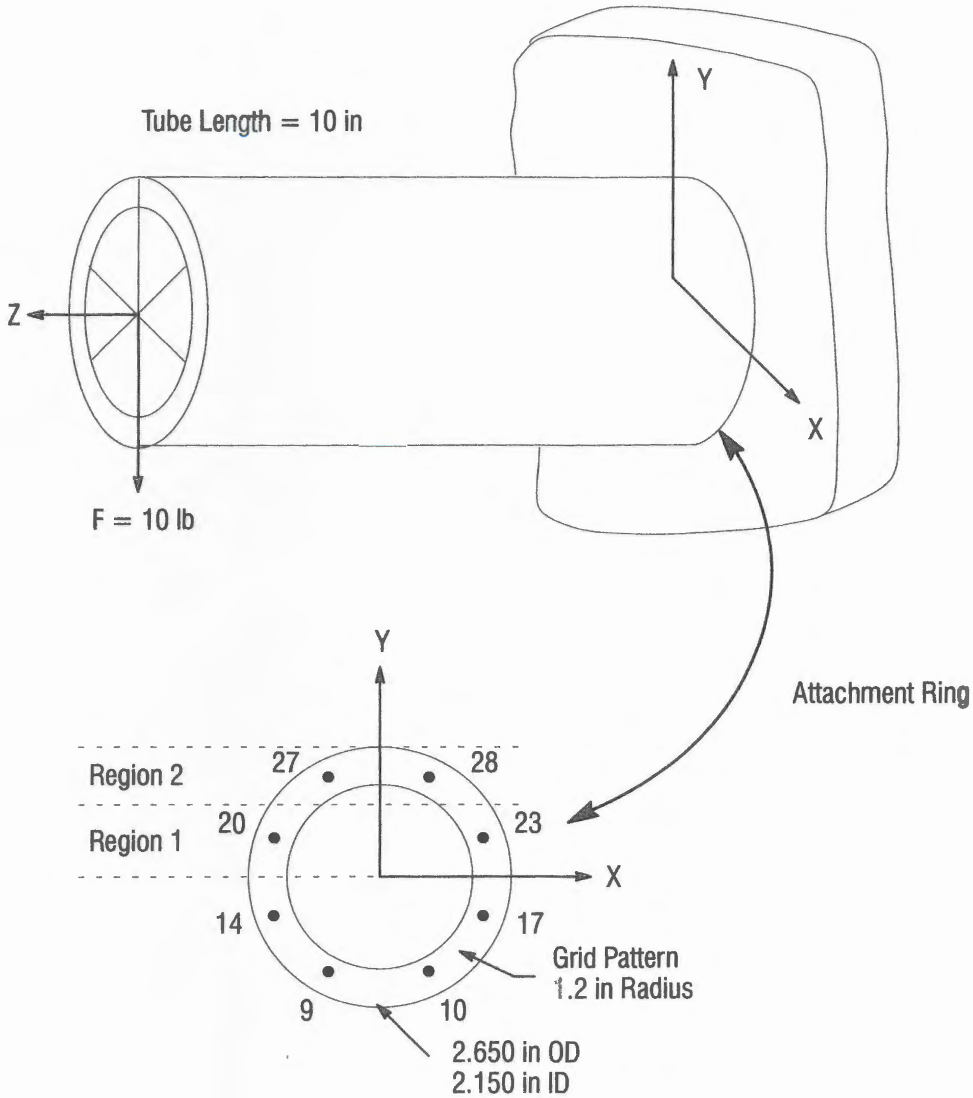


Figure 8-12. Attachment Ring.

The shear force acting on the attachment does not act linearly; it is maximum at the neutral plane and tapers to zero at the top and bottom fibers (if the tube is solid, the shear distribution is a quadratic function, but in our example, it is a thick walled tube). The first step is to calculate the shear forces acting on the attachment ring as a function of the distance from the neutral plane using the classical strength of materials calculations. The result of this calculation is shown in Figure 8-13. The shear force curve is divided into two regions, each region representing a grid point region as shown in Figure 8-12. The area under the curve for each region represents the portion of the shear force transmitted to the grid points within the region. Using these area values as the coefficients for an RBE3 entry, the RBE3 distributes the shear force in a manner similar to the shear force curve.

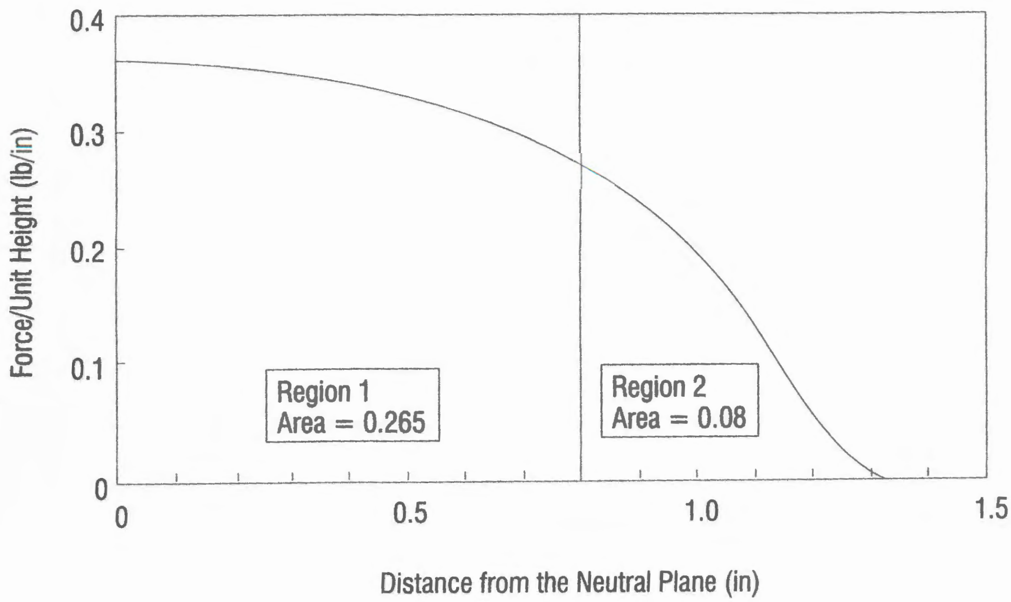


Figure 8-13. Shear Force on the Attachment Ring.

The input file for this example is shown in Listing 8-5. The reaction force and moment due to the applied load of 10 lb acting at the end of the tube are 10 lb and 100 in-lb, respectively.

Listing 8-5. Distributing the Attachment Forces with RBE3.

```
$
$FILENAME - SHEAR1.DAT
$
ID      LINEAR,SHEAR1
SOL     101
TIME    5
CEND
TITLE = SHEAR TEST CASE USING AN RBE3
SET 1 = 9,10,14,17,20,23,27,28
GPFORCE = 1
SPC = 1
LOAD = 1
BEGIN BULK
PARAM   POST    0
PARAM   AUTOSPC YES
$
$ RIGID CONNECTION USING TWO RBE3
$
RBE3    100             99      3456    1.0     123     9       10
        14      17      20      23      27      28
RBE3    101             99      12      0.08    123     9       10
        27      28      0.265   12      14      17      20      23
$
GRID    99              2.0     2.0     0.0
$
FORCE   1       99              1.      0.0     1.0     0.0
MOMENT  1       99              1.      1.0     0.0     0.0
$
$
$ ONLY THE END GRIDS ARE SHOWN
$
GRID    9               1.6     .8      0.0
GRID    10              2.4     .8      0.0
GRID    14              .8      1.6     0.0
GRID    17              3.2     1.6     0.0
GRID    20              .8      2.4     0.0
GRID    23              3.2     2.4     0.0
GRID    27              1.6     3.2     0.0
GRID    28              2.4     3.2     0.0
$
$ QUAD4S, PSHELL, MAT1, AND SPC NOT SHOWN
$
ENDDATA
```

A full model including the model of the tube was also generated for comparison. The grid point forces for the attachment points for both the simple model described above and the full model are summarized in Table 8-3. As can be seen, the force distribution from the RBE3 is close to that of the full model. This approximation is acceptable for many applications. The disadvantage of using this type of simplified model is that the stiffness of the tube is neglected. The full model is located on the delivery media with the name "shear1a.dat".

Table 8-3. Comparison of RBE3 Attachment Forces to the Full Model.

Grid Point	F_Y		F_Z	
	RBE3 Model	Full Model	RBE3 Model	Full Model
8,9	-0.058	-0.077	0.188	0.180
14,17	-0.192	-0.172	0.063	0.074
20,23	-0.192	-0.172	-0.063	-0.074
27,28	-0.058	-0.077	-0.188	-0.180

The most common user error in RBE3 element specification results from placing 4, 5, or 6 in the Ci (independent DOF) field in addition to the translation components. The rotations of the dependent point are fully defined by the translational motion of the independent points. The ability to input 4, 5, or 6 in the Ci field is only for special applications, such as when all of the connected points are colinear.

~ not use 4, 5, or 6 for Ci ~s unless you have a good ~on to do so.

Small checkout models are recommended whenever you are specifying elements with nonuniform weight factors, asymmetric geometry or connected degrees of freedom, or irregular geometry. Using small checkout models is especially necessary when the reference point is not near the center of the connected points.

Use small checkout models when specifying nonuniform weight functions.

In summary, the intended use of the RBE3 element is to transmit forces and moments from a reference point to several non-colinear points. The rotation components 4, 5, and 6 should be placed in the Ci field only for special cases, such as when the independent points are colinear. For further details, see *MSC/NASTRAN Common Questions and Answers*, Second Edition.

Mesh Transition Between Dissimilar Element Types

Special modeling is required because bar and plate elements have rotational stiffness and solid elements do not.

Attaching a plate or bar element to a solid element is a case of transition between dissimilar element types. This process is more involved than it appears at first glance. Solid elements have stiffness only in the translational DOFs at the attachment grid points; they have no stiffness for rotational DOFs. A simple visualization is to think of the attachment of a solid element to a grid point as a "ball-and-socket" joint, that is, translational forces may be transmitted, but no moment may be transmitted through the connection.

This incompatibility of the element stiffness matrices represents a modeling problem whenever plate or bar elements are attached to solid elements. Both plate and bar elements have stiffness for rotational DOFs (although the plate element may not have a stiffness for the normal rotation). Therefore, special modeling must be performed whenever a plate or bar is connected to a solid element. Otherwise, the connection becomes a hinge (for plate elements) or a pinned connection (for bar elements).

Several methods are available to handle the transition between these elements. These methods range from adding extra elements (for example, adding an additional plate or bar that continues into the solid element) to using special (R-type) elements for the transition.

One method of handling this transition is to use RBE3 elements (see Chapter 8). The RBE3 is an interpolation element, which is ideally suited for this application. By using RBE3s, the rotations of the attached grid points is simply slaved to the translations of the adjacent grid
an interpolation element, which is ideally suited for this application. By using RBE3s, the rotations of the attached grid points is simply slaved to the translations of the adjacent grid
points. Examples of using RBE3 elements to connect a solid element to a plate element and bar element are shown in Figure 9-16. The RBE3 elements attach the rotational DOFs to the translational DOFs on the solid element.

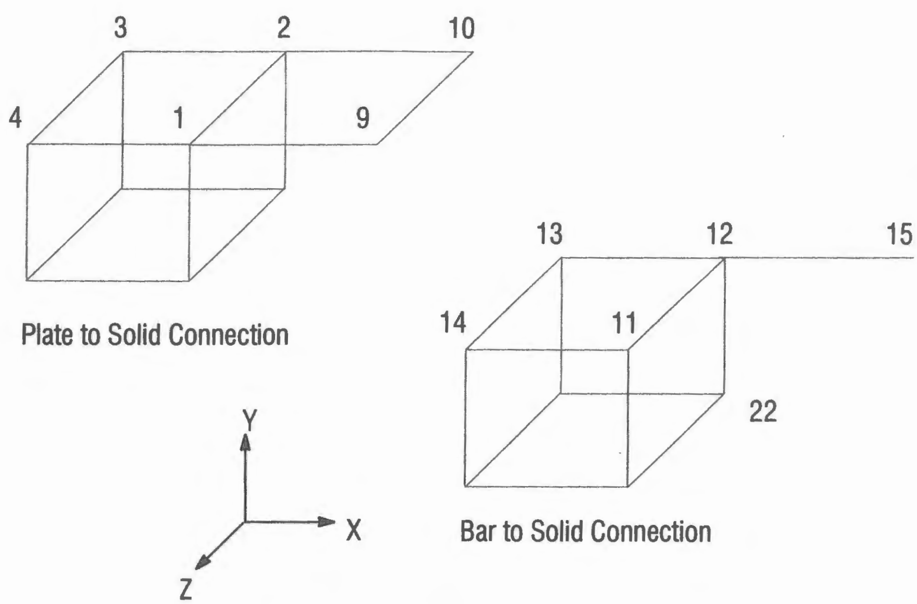


Figure 9-16. Typical Transition Between Dissimilar Elements.

For the plate to solid connection, two RBE3 elements suffice:

1	2	3	4	5	6	7	8	9	10
$RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
RBE3	901		1	456	1.0	123	2	3	
	4								

RBE3 (Plate to Solid Connection)

RBE3	902		2	456	1.0	123	1	3	
	4								

RBE3 (Plate to Solid Connection)

For the bar to solid connection, one RBE3 element can make the connection:

RBE3	903		12	456	1.0	123	11	13	
	22								

RBE3 (Bar to Solid Connection)

These RBE3 elements transmit the loads to the independent DOFs. If RBE2 elements are used, then the connection is "rigid."

The important thing to remember when handling these connections is that the solid elements have no stiffness for rotational DOFs, whereas the real structure does. This means that a special modeling effort is needed when any element with bending stiffness is connected to a solid element.

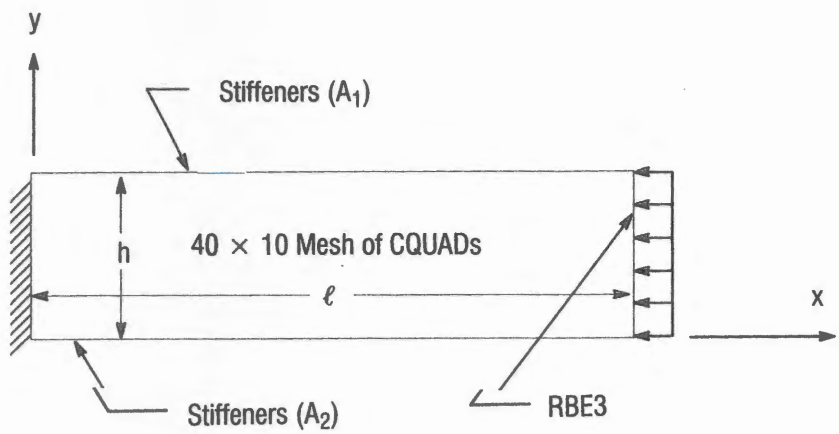
When using the RBE3 element, care must be taken to ensure that the independent DOFs are sufficient to transfer any applied loads. For the bar to solid connection in Figure 9-16, if only two independent grid points are used, the element is "unstable," that is, since only the translational DOFs are used as independent in the sample, the element is unstable for rotation about the axis connecting the two points. Therefore, three non-colinear grid points are used. A simple way to remember this is to ask, "If I constrain the DOFs that I list as independent on the RBE3, can I prevent any possible rigid body motion?" If the answer to this question is "yes," then the RBE3 element is capable of transferring any applied loads. In this way, you can avoid possible problems in processing the RBE3 elements.

The independent DOFs must be capable of representing any general rigid body motion.

EXAMPLE

Example 4 – Buckling of a Stiffened Panel with Transverse Shear Flexibility

This problem is selected to illustrate the effect of the transverse shear flexibility in the buckling failure of a stiffened panel. Figure 13-11 consists of a panel reinforced with stiffeners at both the top and bottom. The panel is subjected to a distributed compressive load at the right edge. The failure mode of interest in this case is the in-plane (xy plane) buckling of the stiffened panel.



$$
\begin{aligned}
A_1 &= A_2 = 0.6\ \text{in}^2 \quad &; \quad h &= 2.0\ \text{in} \\
A_t &= 1.2 + 0.4 = 1.6\ \text{in}^2 \quad &; \quad A_w &= t(h) = 0.2(2.0) = 0.4\ \text{in}^2 \\
E &= 1.0 \times 10^7\ \text{psi} \quad &; \quad I &= 1.333\ \text{in}^4 \\
\ell &= 20\ \text{in} \quad &; \quad G &= 3.79 \times 10^6\ \text{psi} \\
t &= 0.2\ \text{in} \quad &; \quad n &= A_t/A_w = 4
\end{aligned}
$$

Figure 13-11. Buckling of a Stiffened Panel.

MSC/NASTRAN Results for Example 4

The corresponding MSC/NASTRAN input file for this problem is shown in Listing 13-5. The panel is modeled with CQUAD4s. Since only the in-plane buckling failure mode is of interest, only the membrane stiffness is requested for the CQUAD4s (MID1 only). The stiffeners are modeled with CRODs. Since this is a planar model, the out-of-plane motion is constrained (3, 4, 5, and 6 DOFs).

An RBE3 element is used to spread the loads to multiple points.

The panel is subjected to a distributed compressive load at the right end. As an alternative modeling technique, an RBE3 element is connected to the grid points at the right edge so that the load can be applied to a single grid point with the RBE3 spreading the loads to the other grid points.

The SINV method of eigenvalue extraction method is used for this problem. Therefore, the EIGB entry instead of the EIGRL entry is used. Field 3 of the EIGB entry designates the selected method. Fields 4 and 5 provide the range of the eigenvalue of interest. A reduced output showing the eigenvalue table and lowest buckling mode shape is included in Figures 13-12 and 13-13, respectively.

Listing 13-5. Input File for the Buckling of Stiffened Panels.

```
$
$     FILENAME - stffqud4.dat
$
SOL    105
TIME  10
CEND
TITLE = PANEL WITH STIFFENERS
DISP = ALL
STRESS = ALL
SPC = 1
$
SUBCASE 1
LABEL = COMPRESSIVE LOAD
LOAD = 1
$
SUBCASE 2
LABEL = BUCKLING SUBCASE
METHOD = 10
$
BEGIN BULK
$
$EIGRL   10                              1
EIGB    10      SINV    .7              .9
PARAM   POST    0
$
RBE3    1000            1000    123456  2.      12      82      123     +
+       164     205     287     328     369     410     2.      12345   +
+       246     1.0     12      41      451
$
FORCE   1       1000            -1.0    100000.
$
$ THIS SECTION CONTAINS THE LOADS, CONSTRAINTS, AND CONTROL BULK DATA ENTRIES
$
PSHELL  1       1       .2
PROD    2       1       .6
$
MAT1    1       1.+7            .32
$
$ BRING IN THE REST OF THE MODEL
$
include 'stffqud4.blk'
$
ENDDATA
```

			REAL EIGENVALUES			
MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	7.811782E-01	8.838429E-01	1.406680E-01	6.161843E+03	4.813498E+03
2	3	4.946603E+00	2.224096E+00	3.539759E-01	1.096963E+07	5.426240E+07
3	2	5.002625E+00	2.236655E+00	3.559746E-01	1.383715E+04	6.922209E+04

Figure 13-12. Eigenvalues for a Stiffened Panel.

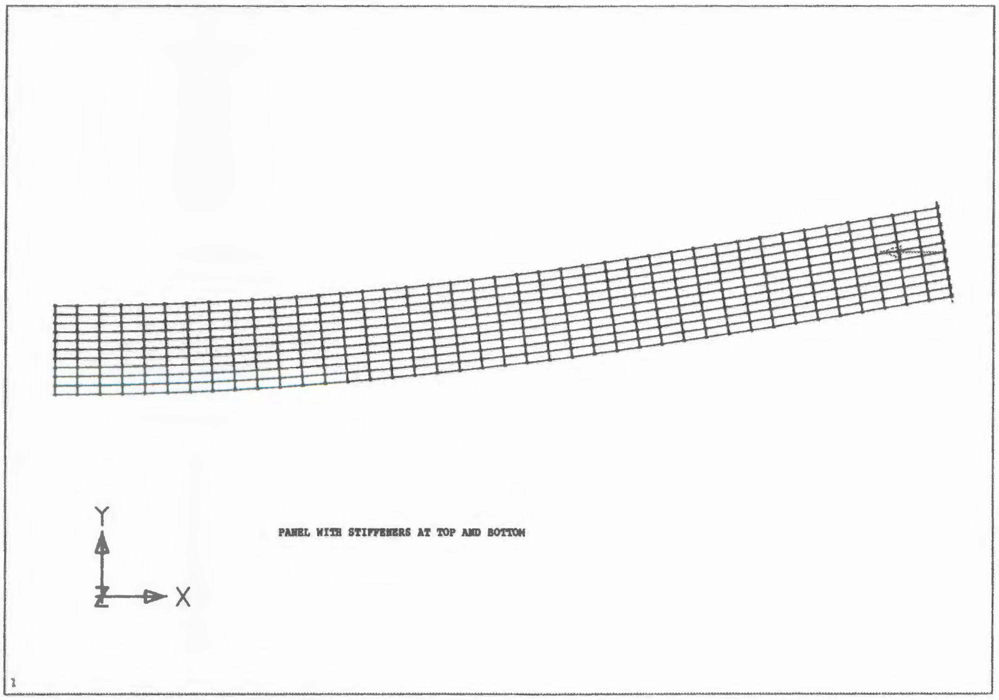


Figure 13-13. Stiffened Panel Buckling Mode.

The lowest buckling load calculated by MSC/NASTRAN is

$$P_{cr_1} = (\lambda_1)(P_a) = 0.78118(-100,000) = -78,\ 118 \text{ lb}$$

Theoretical Results for Example 4

If this problem is treated as an Euler beam, the critical buckling load P_e is equal to

$$P_e = \frac{\pi^2 EI}{4\ \ell^2} = 82,245 \text{ lb} \tag{13-13}$$

If the effect of the transverse shear flexibility is included, then P_{cr} (from Reference 9 in Appendix I) can be calculated as follows:

$$P_{cr} = \frac{\sqrt{1 + \frac{4nPe}{A_t G}} - 1}{\frac{2n}{A_t G}} = 78,210 \text{ lb} \tag{13-14}$$

If the transverse shear flexibility is ignored, then the buckling load deviates by 5.2%.

The transverse shear flexibility is automatically included in MSC/NASTRAN.

Table 13-5 contains a comparison between the theoretical results versus the MSC/NASTRAN results for the buckling of the stiffened panel. Note that the transverse shear flexibility of this structure is automatically included when you are performing a buckling analysis in MSC/NASTRAN. There is no additional input required on your part.

Table 13-5. Stiffened Panel Buckling Results Comparison.

MSC/NASTRAN (lb)	Theoretical (lb)	% Difference
78,118	78,210	0.1

A review of the stresses also indicates that the structure will yield prior to reaching the linear buckling load level. In other words, the critical failure mode may be due to yielding rather than to the linear buckling of the structure. The knowledge of this linear buckling load level can still be of design significance. If you are interested in detailed stresses for this problem, then a nonlinear analysis using Solution 106 may be more appropriate in this case.

Example 5 – Buckling of a Cylinder Under Uniform Axial Load

The next example is the buckling of a cylinder subjected to a distributed compressive load at one end and simply supported at the other end. The cylinder has a diameter of 20 inches and a length of 20 inches with a wall thickness of 0.03 inches as shown in Figure 13-14. This problem illustrates the phenomenon of the buckling of a thin curved shell structure.

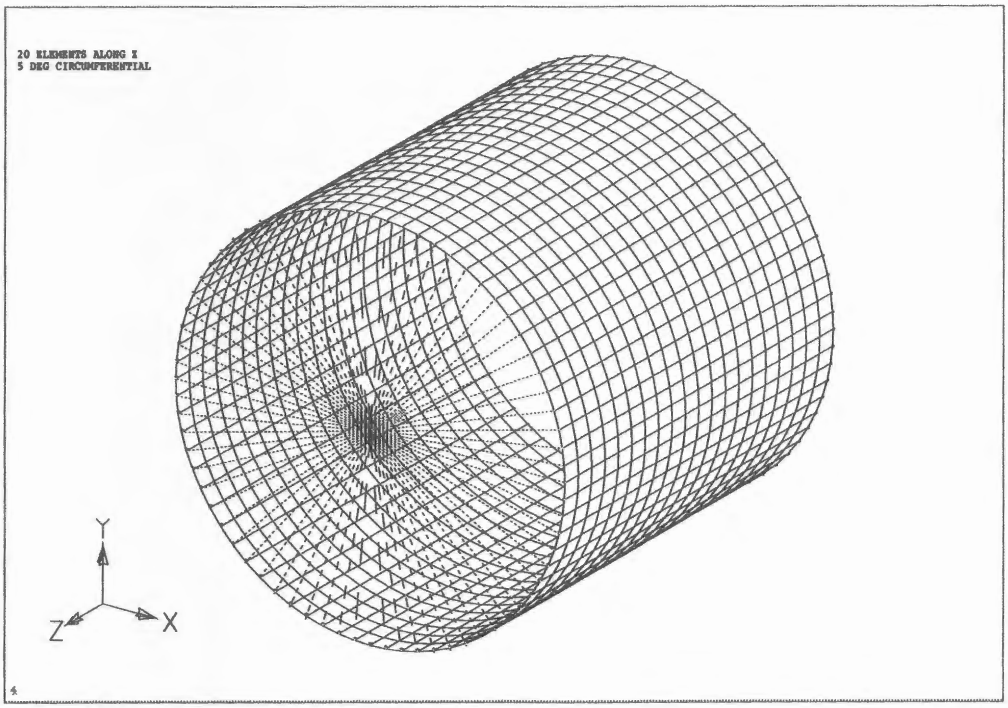


Figure 13-14. Buckling of a Cylinder Under a Compressive Load.

MSC/NASTRAN Results for Example 5

The CQUAD4 elements are used for the MSC/NASTRAN model. A mesh of 20 elements along the length and 72 elements around the circumference is used for this problem. The corresponding input file containing the pertinent entries is shown in Listing 13-6. Once again, an RBE3 element is used to distribute the load from a single grid point to the circumference of the cylinder.

A static load of -100,000 lb is applied in Subcase 1 to generate the internal forces for the structure. This load is applied to the RBE3, which, in turn, distributes the loads around the circumference of the cylinder. The static deflection is shown in Figure 13-15.

The Lanczos eigenvalue extraction method is used in this case to request the lowest mode. The lowest calculated eigenvalue is equal to 0.35854. The buckling load is, therefore, equal to

$$P_{cr} = (.35854)\,(\text{-}100,000) = \text{-}35,854 \text{ lb}$$

A minimum of five grid points per half sine wave is recommended.

The corresponding buckling mode shape is shown in Figure 13-16. As you can see from the plots, there are two grid points per half sine wave (four grid points per sine wave), which is below the recommended value of a minimum of five grid points per half sine wave.

This problem is then remeshed with 40 elements along the z-direction keeping the same mesh density in the circumferential direction. The new eigenvalue calculated in this case is equal to 0.34134. The revised buckling load is, therefore, equal to

$$P_{cr} = (.34134)\,(\text{-}100,000) = \text{-}34,134 \text{ lb}$$

The corresponding mode shape is shown in Figure 13-17. In this case, there are three grid points per half sine wave (six grid points per sine wave), which is still below the minimum requirement. To obtain better accuracy, you can certainly further refine this model until you meet the minimum number of grid points requirement. However, this mesh is sufficient for our goal of demonstrating the linear buckling features of MSC/NASTRAN.

Listing 13-6. Input File for a Cylindrical Buckling Problem.

```
$
$  FILE - buckcy20.dat
$
ID CYLIN BUCKLING
SOL    105
TIME  200
CEND
TITLE = BUCKLING OF CYLINDER - SIMPLY SUPPORTED
SUBTITLE = 20" x 20" -  t=.03" - K6ROT= 10000.
DISP = ALL
SPC = 1
$
SUBCASE 1
LABEL = STATIC LOAD
LOAD = 1
SPCF = ALL
$
SUBCASE 2
LABEL = EIGENVALUE CALCULATION
METHOD = 1
$
BEGIN BULK
$
PARAM   K6ROT   10000.
PARAM   POST    0
PARAM   AUTOSPC YES
EIGRL  1                    1
$
PSHELL  1       1       .03     1
$
MAT1    1       1.+7            .3
$
RBE3    5000            5000    123456  1.      123     381     382     +
+       383     384     385     386     387     388     389     390     +
+       391     392     393     394     395     396     397     398     +
+       399     781     782     783     784     785     786     787     +
+       788     789     790     791     792     793     794     795     +
+       796     797     798     1180    1181    1182    1183    1184    +
+       1185    1186    1187    1188    1189    1190    1191    1192    +
+       1193    1194    1195    1196    1197    1579    1580    1581    +
+       1582    1583    1584    1585    1586    1587    1588    1589    +
+       1590    1591    1592    1593'   1594    1595
$
FORCE   1       5000    0       100000.                 -1.
$
$   INCLUDE THE REST OF THE BULK DATA ENTRIES
$
include 'full2.blk'
$
ENDDATA
```

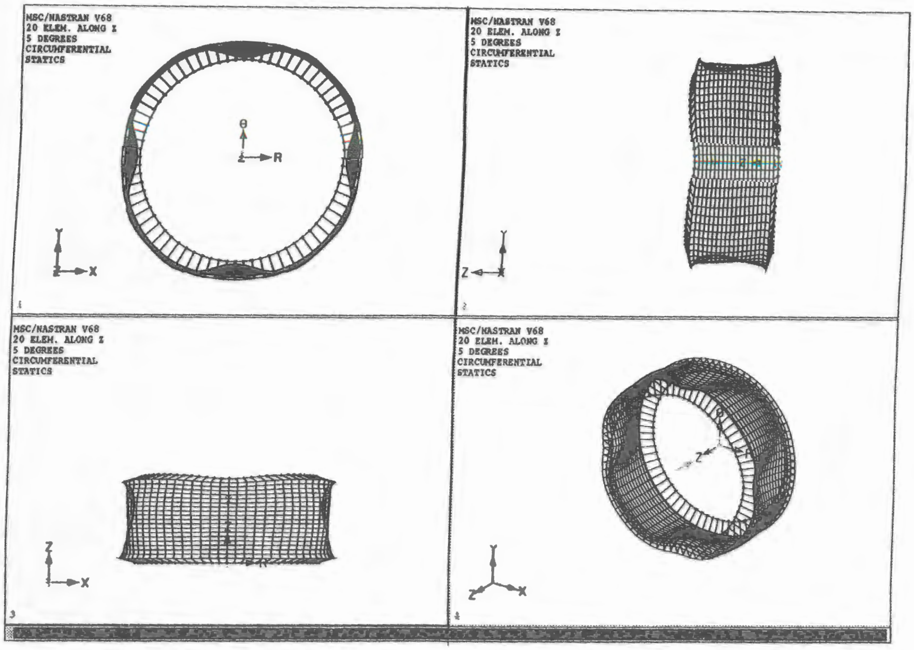


Figure 13-15. Static Deflection of a Cylinder Due to a Compressive Load.

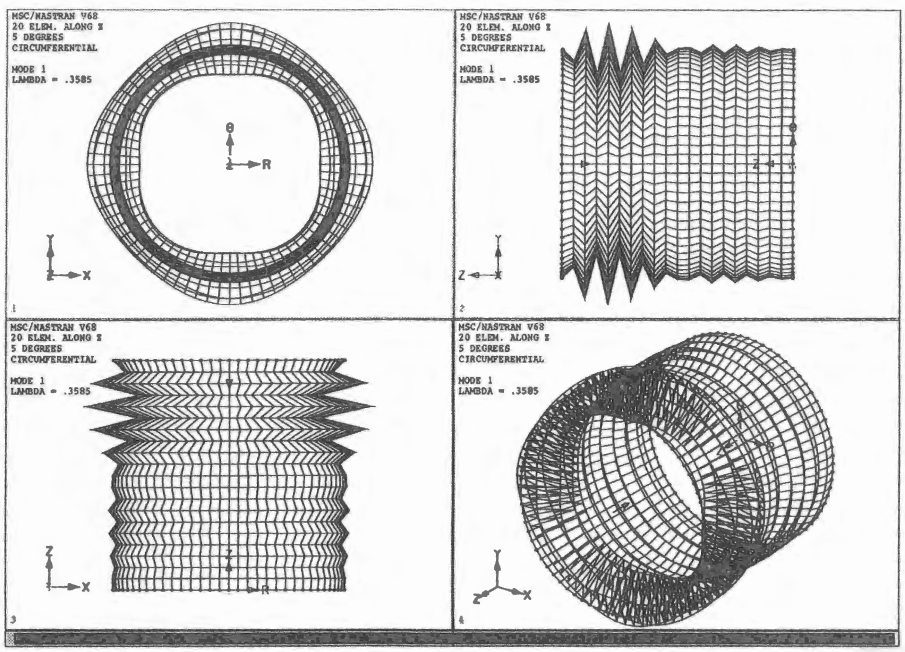


Figure 13-16. Buckling Shapes of a Cylinder with 20 Elements Along the z-Direction.

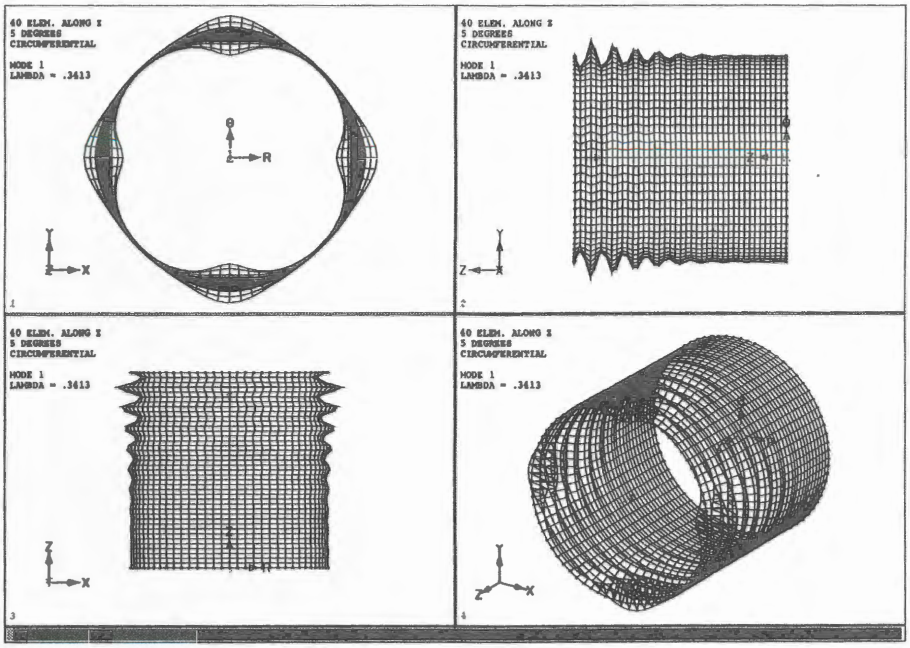


Figure 13-17. Buckling Shape of a Cylinder with 40 Elements Along the z-Direction.

Theoretical Results for Example 5

The first buckling load (Reference 9) can be calculated as follows:

$$Nx = \frac{E(h)^2}{r\sqrt{3(1-\nu^2)}} = \frac{1x\ 10^7\ (.03)^2}{10\ \sqrt{3(1-.3^2)}} = 544.7\ \text{lb/in}$$

$$P_{cr} = N_x\ (2)\ (\pi)\ (r) = 544.7\ (2)\ (\pi)\ (10) = 34,225\ \text{lb}$$

Table 13-6 contains a comparison between the results obtained with MSC/NASTRAN versus the theoretical results.

Table 13-6. Thin Cylinder Buckling Results Comparison.

Mesh Density	MSC/NASTRAN (N)	Theoretical (N)	% Difference
72 × 20	35,854	34,225	4.76
72 × 40	34,134	34,225	0.26

Note that the cylinder is an axisymmetric structure; therefore, you can certainly take advantage of this symmetry feature by modeling a strip of the cylinder instead of the whole model. In this case, you would use Solution 77 instead of Solution 105. Solution 77 is known as the cyclic buckling solution sequence. The use of cyclic symmetry is certainly more efficient from a computational standpoint—at the expense of slightly more complicated user interaction. The disadvantage is that you cannot visualize the behavior of the structure the way that you can with a full model. See Chapter 16 for further details regarding the subject of cyclic symmetry.

Example 6 – Multiple Buckling Analyses in a Single Run

So far a single static analysis was considered followed by a single buckling analysis. Example 6 shows you how to run multiple static and buckling analyses in a single run. In fact, what is done in this example is to combine the Euler beam buckling (Example 1) and lateral buckling (Example 2) problem into a single run. The detail input file is shown in Listing 13-7.

The model used is the same geometric beam model used in Examples 1 and 2. The first subcase (Subcase 2) is a static subcase consisting of a cantilever beam with a vertical tip load applied to the free end (see Figure 13-6). The second subcase (Subcase 5) is a static subcase consisting of a simply supported beam with an axial load applied to the roller end (see Figure 13-2). The third subcase (Subcase 11) is for a lateral buckling analysis. The Case Control command (STATSUB = 2 in Subcase 11) tells MSC/NASTRAN that you want to generate the differential stiffness matrix from the first static subcase (Subcase 2). The fourth subcase (Subcase 21) is for a Euler beam buckling analysis. The Case Control command (STATSUB = 5 in Subcase 21) tells MSC/NASTRAN that you want to generate the differential stiffness matrix from the second static subcase (Subcase 5). The Bulk Data entries are similar to Examples 1 and 2. Note that different boundary conditions are allowed for different subcases.